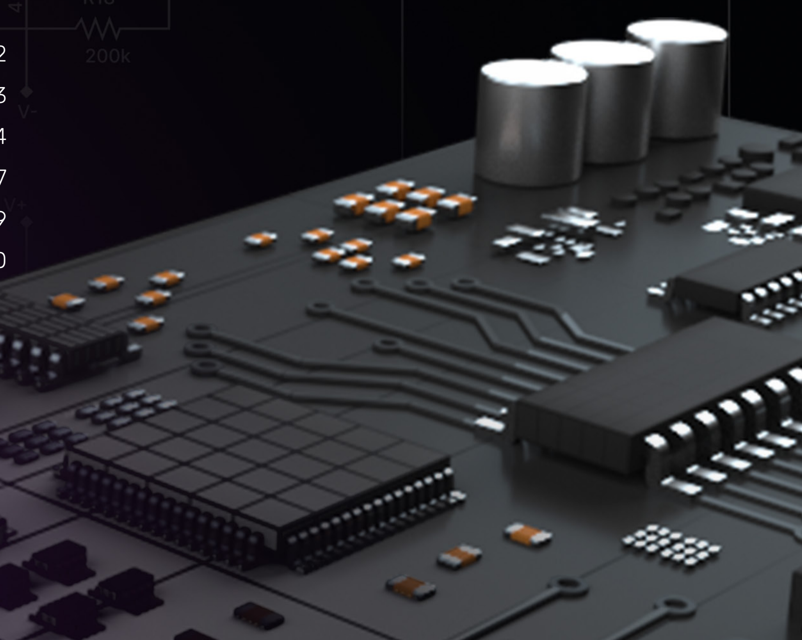
MIGRATION **GUIDE**

cādence

Migrating from PADS to OrCAD

Contents



Overview

Choosing the right PCB design solution is never an easy task. No matter if you are a startup company looking for tools to develop your next innovative electronic product or a large enterprise wanting a better solution to improve the productivity of your design team, selecting a PCB solution can be a daunting task. No one wants to get 75% of the way through a design to find out that the software you selected is not going to achieve what you need to accomplish.

Before you select a PCB design software package, there are many performance and capability aspects you should consider first:

- Does the capabilities of the application and its technology meet your design requirements?
- Does the design software licensing fit within your budget?
- What level of support can you expect? Will you be able to get quick responses to your questions and access online tutorials? Is local help available?
- Can the application scale with your needs? As designs are getting more and more complex, will the capabilities of the tool adjust accordingly?
- How many other companies in your industry are using this tool and what is their feedback?

OrCAD® offers an excellent solution for individual designers, small design teams, and large enterprises. OrCAD offers constraint-driven design, advanced auto/interactive routing, high-speed design, DFM, dynamic shape technology, and much more, helping you deliver high-quality, first-time-right designs in the shortest timeframes. You can be confident that you will have the right solution and technologies at an affordable price to meet all of your design challenges today and tomorrow. Here are five of many reasons why:

- 30 years of innovation and leadership in the industry
- Affordable price and flexible purchase models
- Cutting-edge technologies
- Ecosystem empowered
- Industry's best customer support

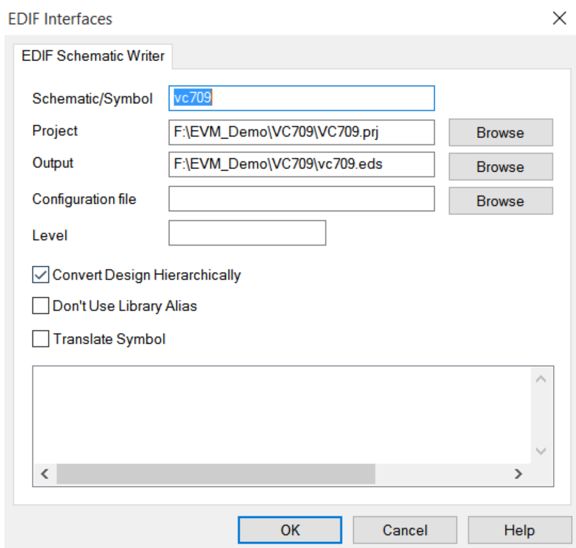
Cadence and OrCAD provide the only full scalable PCB design solution on the market that can seamlessly grow with your needs. OrCAD products are backed by Cadence and their network or certified Cadence Channel Partners (CCP). Get help when you need it by phone or email from local, knowledgeable PCB design professionals.

Like many companies selecting OrCAD, you have existing or legacy designs you need to convert or translate into OrCAD. The good news is that OrCAD is supplied with an integrated and proven PADS translator built in. This guide will walk you through the steps and process involved in getting your design IP into the OrCAD format so you can start realizing the advantages of moving to OrCAD!

Import Your PADS Schematic Data to OrCAD Capture

Step 1 – Export Schematic Data from PADS

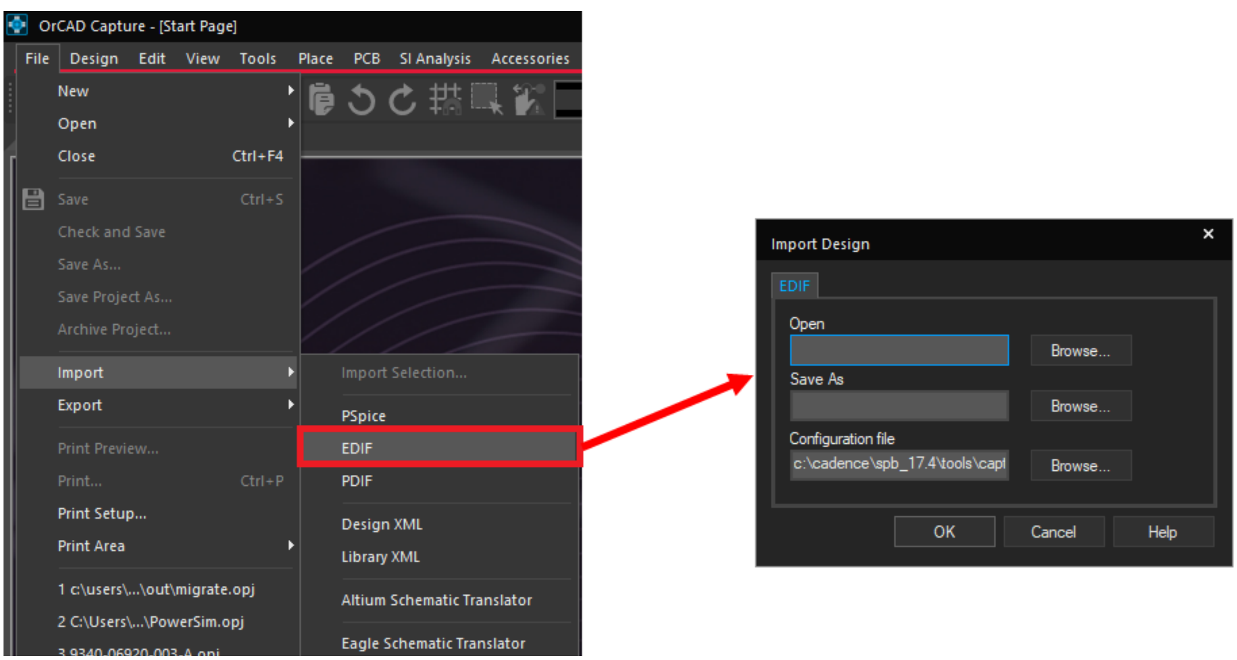
In DxDesigner export your PADS schematic to EDIF 2.0 format using the EDIF Interfaces window:



Note: If you are using PADS Logic you will need to convert your design into DxDesigner first using the Mentor-supplied conversion utilities, then you can proceed with Step 1 of the conversion process.

Step 2 – Import Schematic Data to OrCAD Capture

Once the EDIF file is created, launch OrCAD Capture and select the **"File » Import » EDIF"** command from the top menu.



Select your EDIF file, your destination output folder, and make sure you select the correct configuration file. Click **OK** to begin the translation. Once the EDIF file has translated into OrCAD Capture, some text cleanup may be required.

Import Your PADS PCB Design into OrCAD PCB Editor

Step 1 – Preparation

In the PADS PCB application, export your PCB database to the most recent ASCII format. The newer version of the ASCII can be exported using PADS version 5 or later by choosing the menu items: "**File » Export » ASCII**"

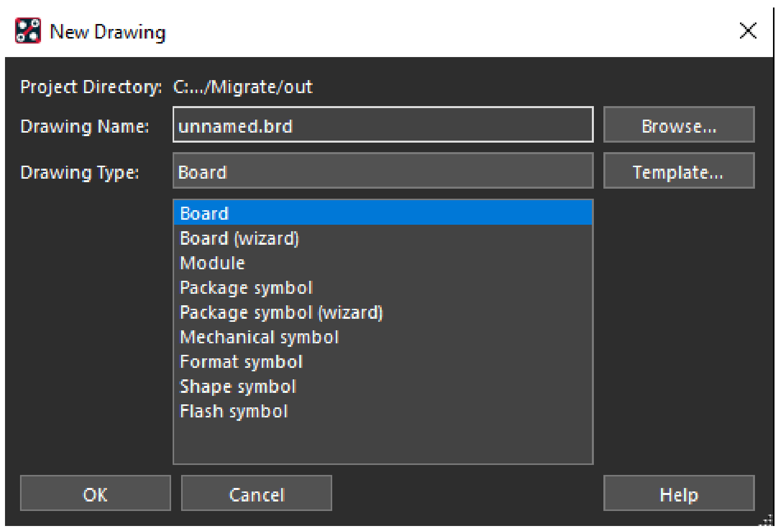
This creates an ASCII database (*.ASC) that will be used for translation into OrCAD Capture format. Refer to the PADS documentation if more detail on exporting an ASCII database is required.

Note: Only copper pours in PADS defined as positive data are recognized by the translator.

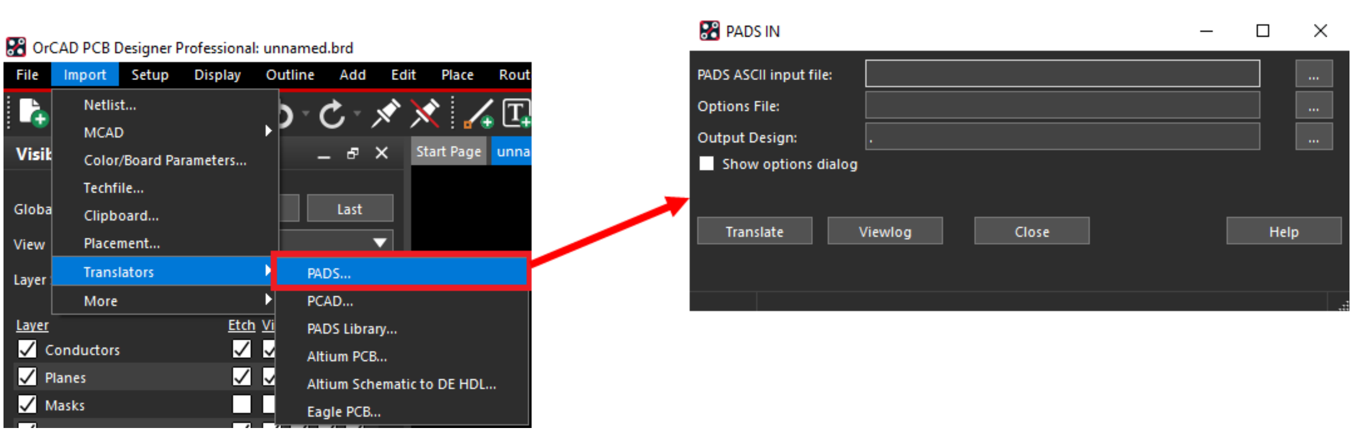
Step 2 – Create a Blank Board

This step will ensure the translated PCB file is placed in the correct location. Create a new board from the home screen in the OrCAD PCB Editor. You can also use the **"File » New…"** command from the top menu bar.

In the New Drawing window, navigate to the folder where you want the translated PCB to be saved. Enter a board name; this name is just a placeholder and can be changed after the import completes.



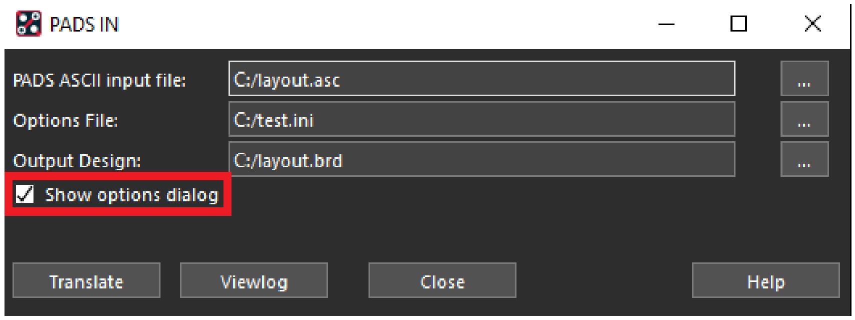
In OrCAD PCB Editor, choose **"Import » Translators » PADS"** from the top menu bar:



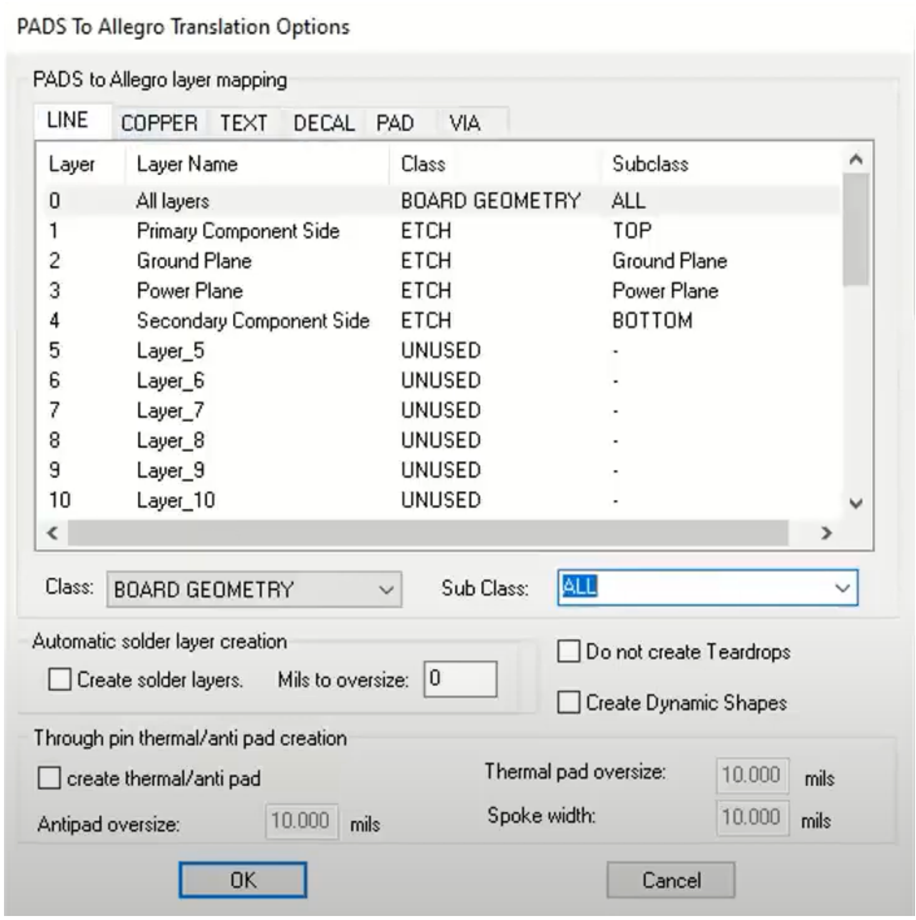
Browse to the **.ASC** file to be translated, create an options file, and select an Output Design folder.

Step 3 - Layer Mapping to OrCAD PCB Editor

Select your ASCII database for Import, enter an options file name and select "**Show options dialog**" to create your options file, then select **"Translate"**:



Map PADS layer numbers to the corresponding OrCAD PCB Editor layers (PCB footprint, board data layers, etch):

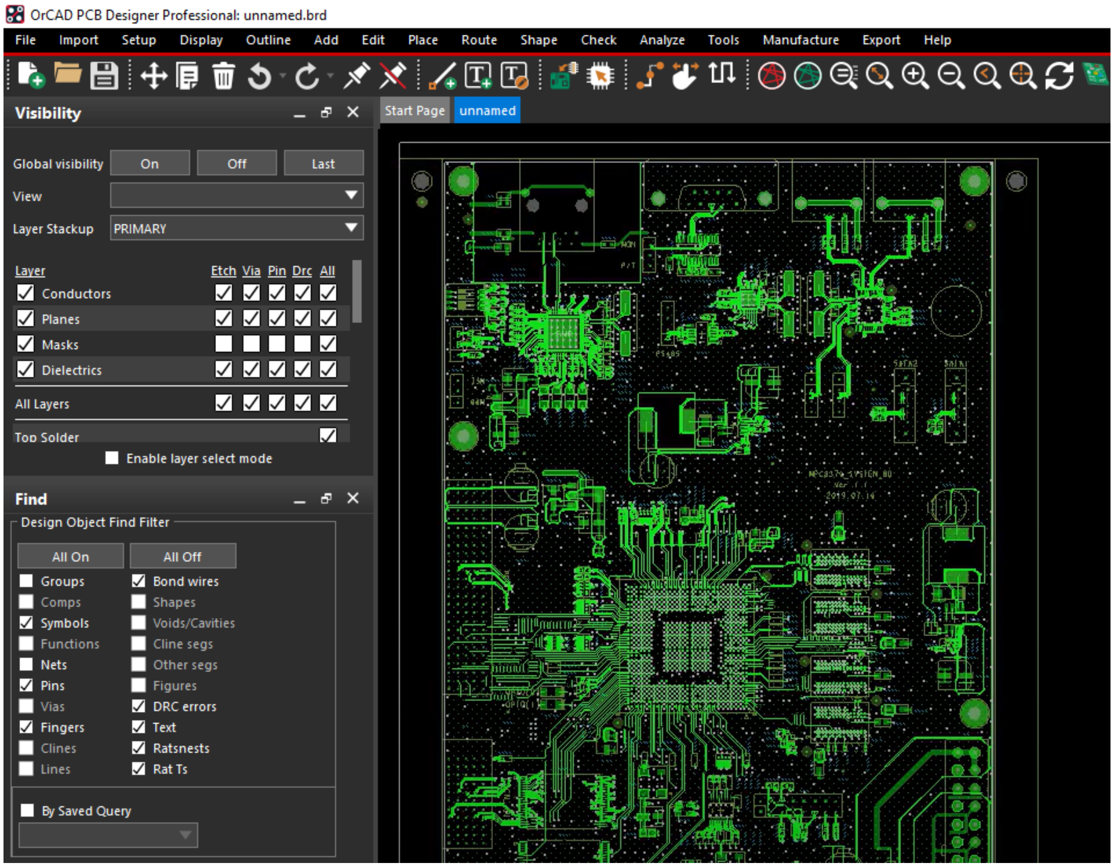


After mapping, the Command window will show the translation progress. Once the translation finishes, the translated **.BRD** file will be accessible in the folder specified in the Output Design field.

Note: In some cases, especially for larger boards, it might seem like PCB Editor has stopped working. Do not close it as tests on large boards have shown translation times taking over five minutes, although this is unusual.

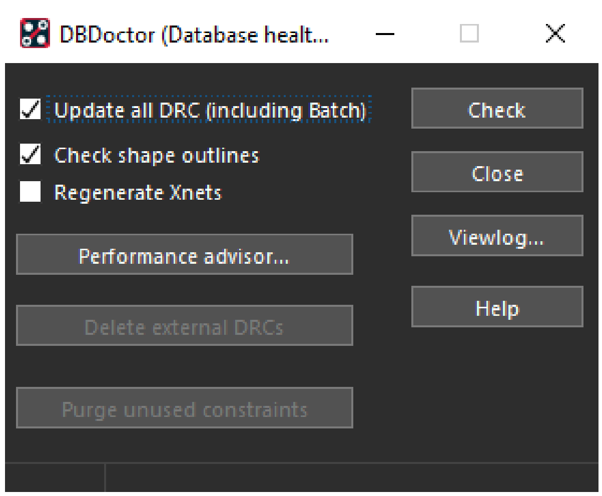
Note: Make sure to check the log file after the translation completes. The log file will contain all the text shown in the Command window once the translation completes; this file can be found in the root folder as the translated **.BRD file**.

The translated PCB layout will appear in the main editor window inside the new drawing that was created in Step 2. Save the **.BRD file** before continuing.



Step 4 - Checking the Imported Database for Design Errors

Check the database for errors using the command **"Check » Database Check"**:



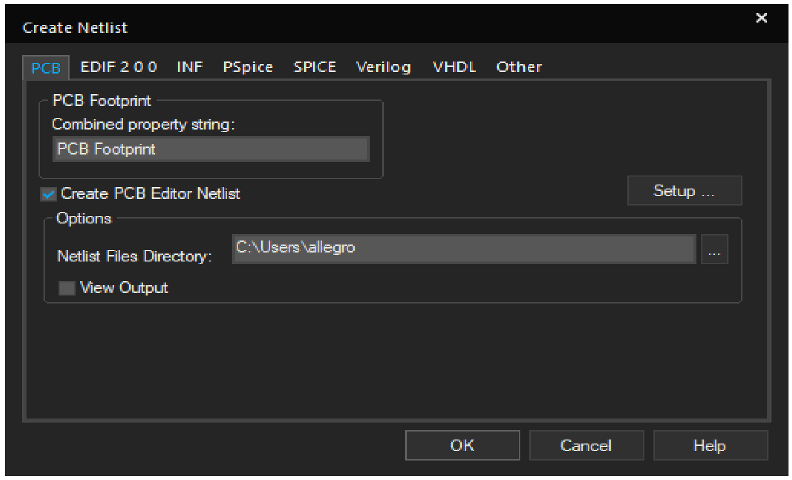
The DBDoctor will flag any errors or remaining DRC errors that should be addressed during cleanup.

Synchronize the Schematic to the Migrated PCB

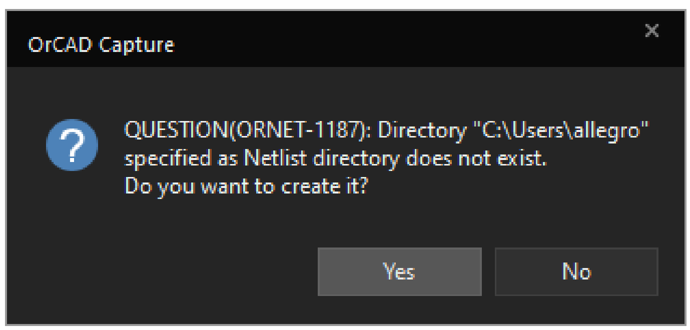
Step 1 – Creating a Netlist in OrCAD Capture

From OrCAD Capture, open the **.OPJ** file with the translated schematics.

Select **"Tools » Create Netlist"**.



Click **"OK"** to create the netlist and accept saving the project and creating an **allegro** subdirectory.



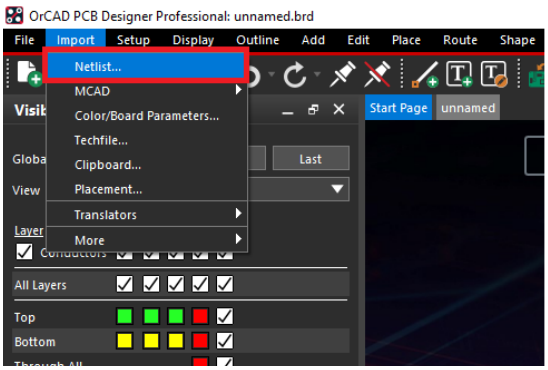
Note: The **allegro** subdirectory with the netlist is created in the directory where the OrCAD Capture design is located. This directory name will be included by default. If you change this target directory, you will need to make sure you point to this new directory when you import the netlist in Step 2.

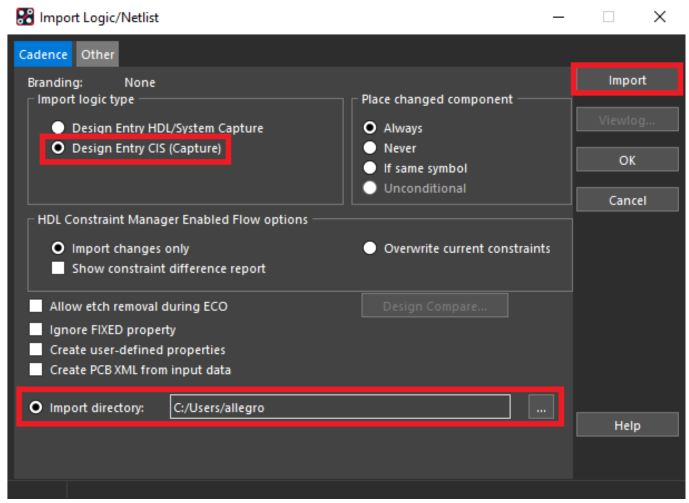
Note: Duplicate pin names (other than PWR or GND designated pins) must be resolved. Logic symbols and PCB footprint pin mismatches may need to be corrected. (This occurs when unused pins on the logic symbols are not accounted for with no-connect property.)

Once the schematic has been synchronized to the translated PCB, cross-probing and cross-placement tools can be utilized.

Step 2 – Importing the Netlist in OrCAD PCB Editor

Select “**Import » Netlist”.**





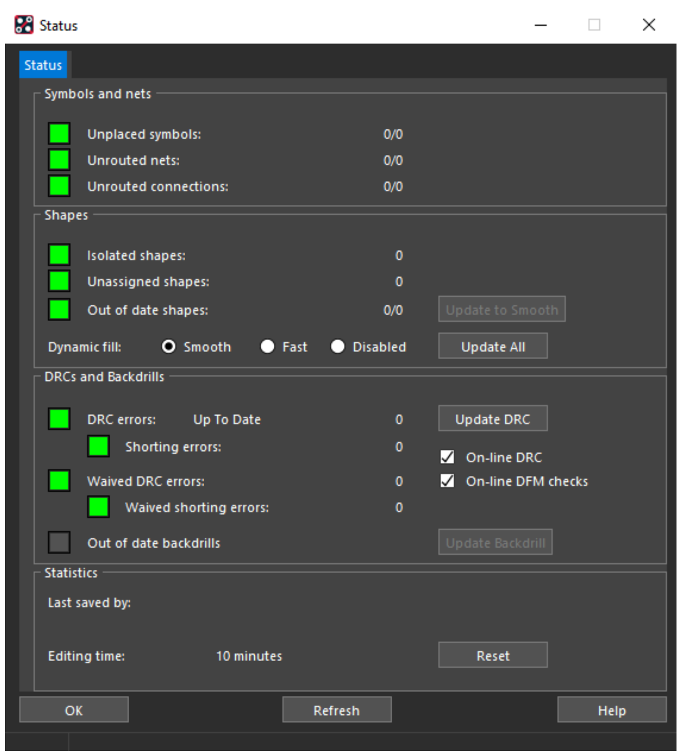
In the Import Logic window, make sure that the Import directory points to the **allegro** subdirectory with the netlist. This directory was created in the previous step.

Note: If you used a different directory name in Step 1, then make sure you navigate to that directory in the Import Logic window.

Click **“Import”** to import the netlist.

Step 3 – Check Design Status

Select **“Check » Design Status”** and, if necessary, click **“Update to Smooth”** to update the dynamic copper shapes.



Step 4 – Check Physical and Spacing DRC Constraints

From OrCAD Capture, start the Constraint Manager from the **"PCB » Constraint Manager"** menu item. Check and verify the physical and spacing DRC rules.

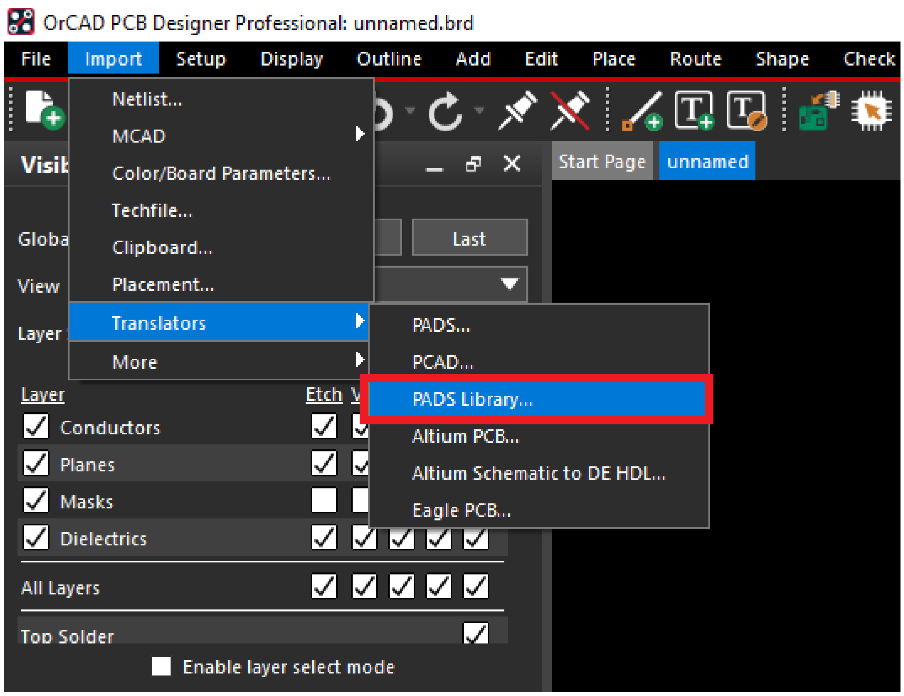
Update the online DRC using the toolbar icon.

Step 5 – Change to Preferred Colors

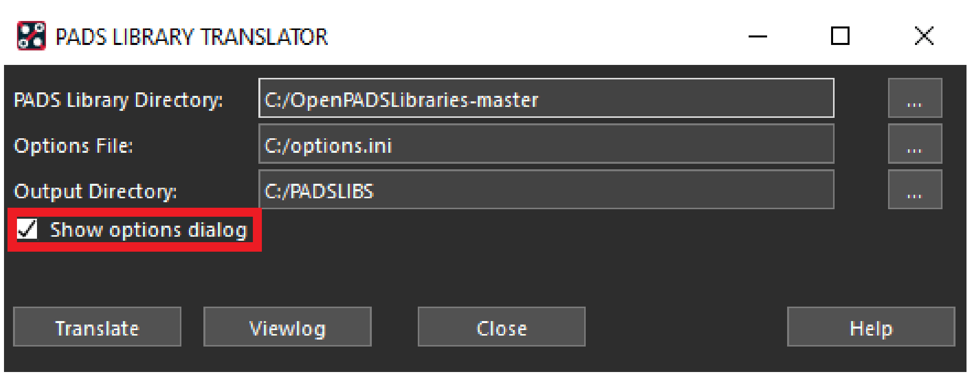
Changing the colors to suit your preferences is easy. Select the color and visibility toolbar icon and change your color preferences accordingly.

Importing Your PADS PCB Libraries into OrCAD PCB Editor

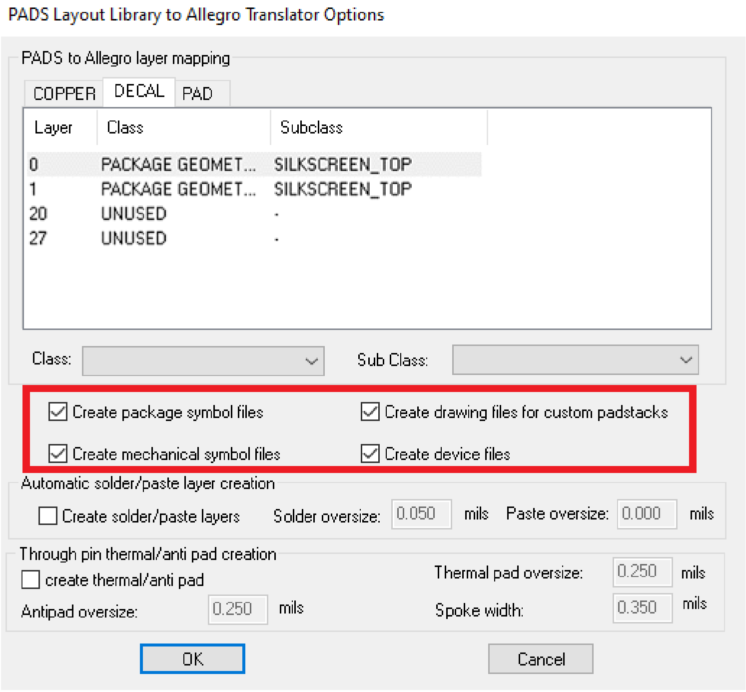
Invoke the library translator from within OrCAD PCB Editor using the **"Import » Translators » PADS Library…"** command from the top menu.



Use the OrCAD library translator to browse to the folder where your PADS package and decal libraries are located, and specify your desired output folder for the migration. Type in the file name for the options file you wish to create, select the "**Show Options Dialog"**, and then **"Translate"**:



Select the options below:



The OrCAD PCB Editor library footprints and padstacks will be available in the specified folder as **.DRA files.**

Next Steps

Now that you know how easy it is to move to OrCAD PCB Designer, are you ready to learn more about the exciting OrCAD features and technologies which will help you improve your design productivity? Here are some resources you can leverage to learn more about OrCAD technologies.

What's New in OrCAD

Want to know what are the new features in the latest OrCAD release? Check out what's new.

Customer Testimonials

See how companies leverage OrCAD to bring their products to market on time and budget. Read OrCAD customer stories.

Product Information

Need more videos, application notes, or datasheets to dive deeper into the OrCAD technologies? View OrCAD product pages.

If you have any questions about migration or OrCAD, please do not hesitate to contact your local Cadence Channel Partner at http://orcad.com/about/contact-us.

cādence®

Cadence is a pivotal leader in electronic design and computational expertise, using their Intelligent System Design Strategy to turn design concepts into reality. Cadence customers are the world's most creative and innovative companies, delivering extraordinary electronic products from chips to boards to systems in the most dynamic market applications. **www.cadence.com**